

Large-Eddy Simulation in Complex Domains using the Finite Element Method

Rose McCallen and Barbara Kornblum

Lawrence Livermore National Laboratory

University of California

Livermore, California

Wolfgang Kollmann

Department of Mechanical and Aeronautical Engineering

University of California, Davis

Davis, California

ABSTRACT

The analysis of three-dimensional (3D), transient, turbulent fluid flows that exhibit complicated vortex shedding patterns is addressed. Many engineering flows of practical interest possess these characteristics, and familiar examples include the flapping vortical flow around moving vehicles, bridges and buildings, and flow in pipe bends, ducts, and channel expansions. Large-eddy simulation (LES) of these flows provides the engineer with three dimensional and time dependent velocity and pressure fields. LES is a turbulence modeling technique that calculates the large-scale motion explicitly (i.e., the resolved field) and the small-scale motion is modeled (i.e., approximated with semi-empirical relations).

In this paper, we demonstrate that the finite element method (FEM) in combination with LES provides a viable tool for the study of turbulent, separating channel flows, specifically the flow over a 3D backward-facing step. This work is an extension of our previously established two-dimensional (2D) simulations to 3D (McCallen et al. 1994). The combination of the LES and FEM methodologies brings together the advantages of each: LES provides a high degree of accuracy with a minimum of empiricism for turbulence modeling and FEM provides a robust way to simulate flow in complicated domains of practical interest.

The flow fields are calculated using a finite element spatial discretization of the incompressible Navier-Stokes equations. With our FEM approach, the discrete pressure Poisson equation is solved, so that continuity and momentum are decoupled and an explicit time-integration scheme is used. The Q1P0 element formulation is used which provides bilinear velocity support in 2D and trilinear support in 3D with piecewise constant pressure. For the backward-facing step, no-slip boundary conditions are imposed on the step and channel walls, a flat velocity profile is specified at the channel inlet, and zero natural boundary

conditions are imposed at the channel outlet. For the 3D simulations, periodic boundary conditions are enforced on the lateral boundaries.

The wealth of data generated in our LES is evaluated by several data analysis methods. The evolution to a 3D state and the approximate periodic nature of the turbulent backward-facing step is demonstrated by flow visualization as well as analysis techniques used to evaluate the velocity and pressure time histories. The benefits and complementing characteristics of each analysis method are demonstrated. Flow visualization clearly captures the significantly different vortex shedding patterns predicted for 2D versus 3D simulations for the backward-facing step. For example, in 2D, the vortices shed from the step and in 3D they shed from the step recirculation zone. We demonstrate how the power spectrum from the velocity time histories at points in the flow indicates the dominant frequencies and hence, the length scales in the flow. This information can be used to evaluate both spatial and time step convergence. As is typically done, we also present the time-average solution for the backward-facing step and describe our code algorithms for calculating these averages during runtime. With this approach, less disk storage for the results is required.

References

McCallen, R.C., Gresho, P.M., Leone, J.M., Kollmann, W., 1994, "Large-Eddy Simulation using the Finite Element Method," Proceedings of the ASME FED Summer Meeting, Incline Village, Nevada, June 19-23, 1994.

This work was performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under Contract W-7405-Eng-48.